



Lesson1: Getting Start

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FIBO



What is SolidWorks?






- **SolidWorks is design automation software.**
- **In SolidWorks, you sketch ideas and experiment with different designs to create 3D models.**
- **SolidWorks is used by students, designers, engineers, and other professionals to produce simple and complex parts, assemblies, and drawings.**

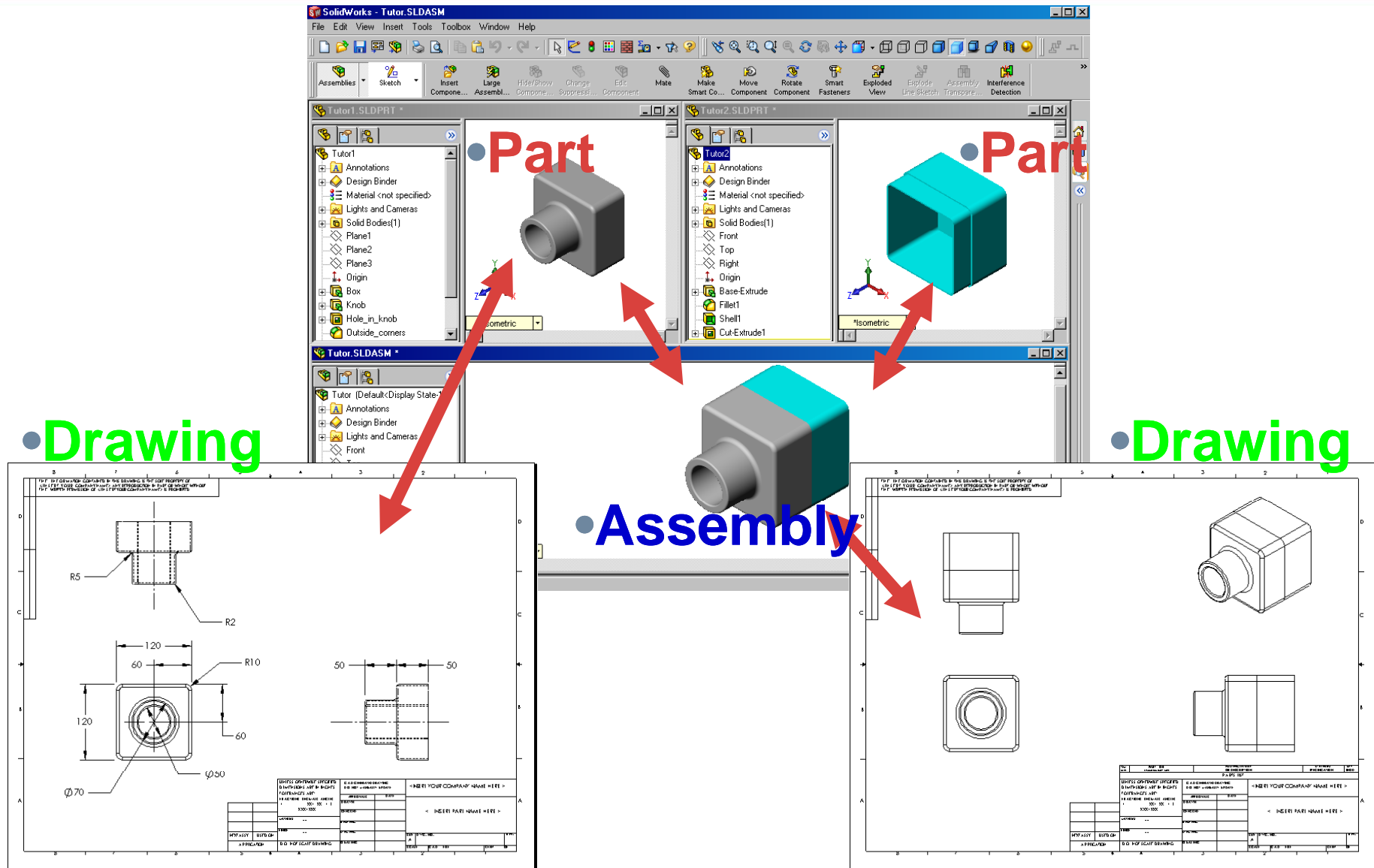
The SolidWorks Model



- **The SolidWorks model is made up of:**

-  **Parts**
-  **Assemblies**
-  **Drawings**

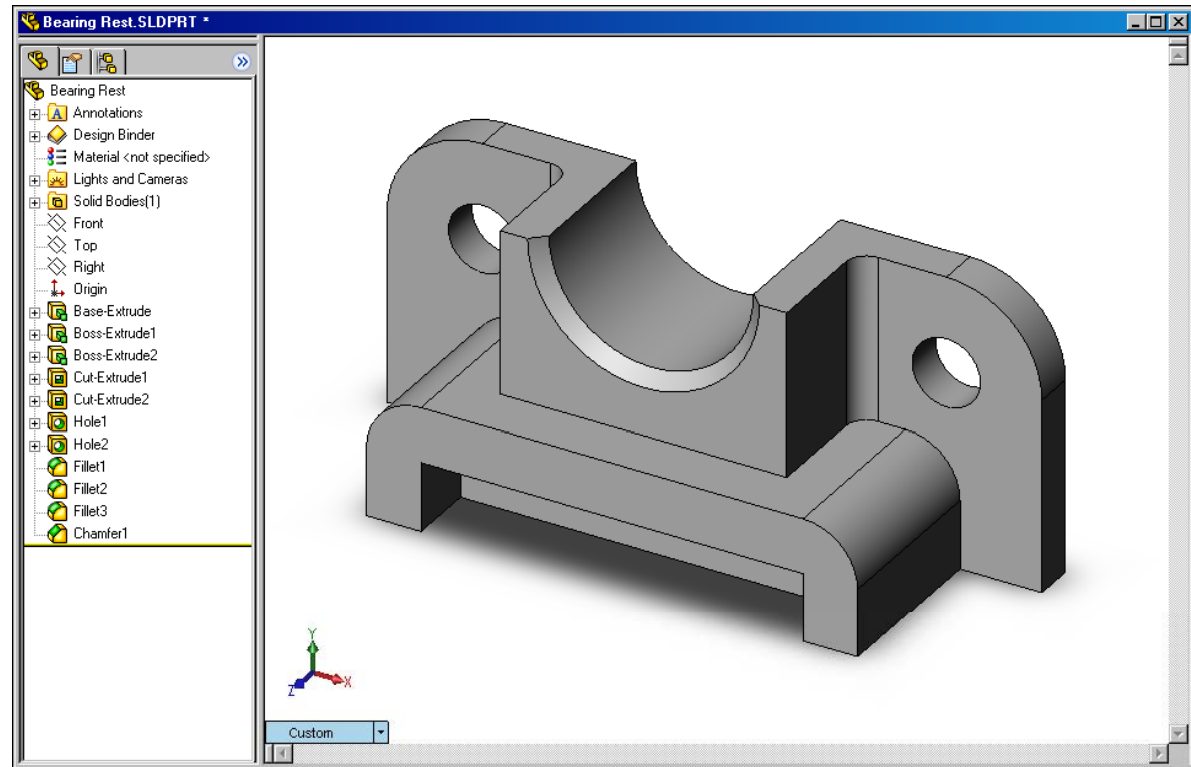
The SolidWorks Model



Features



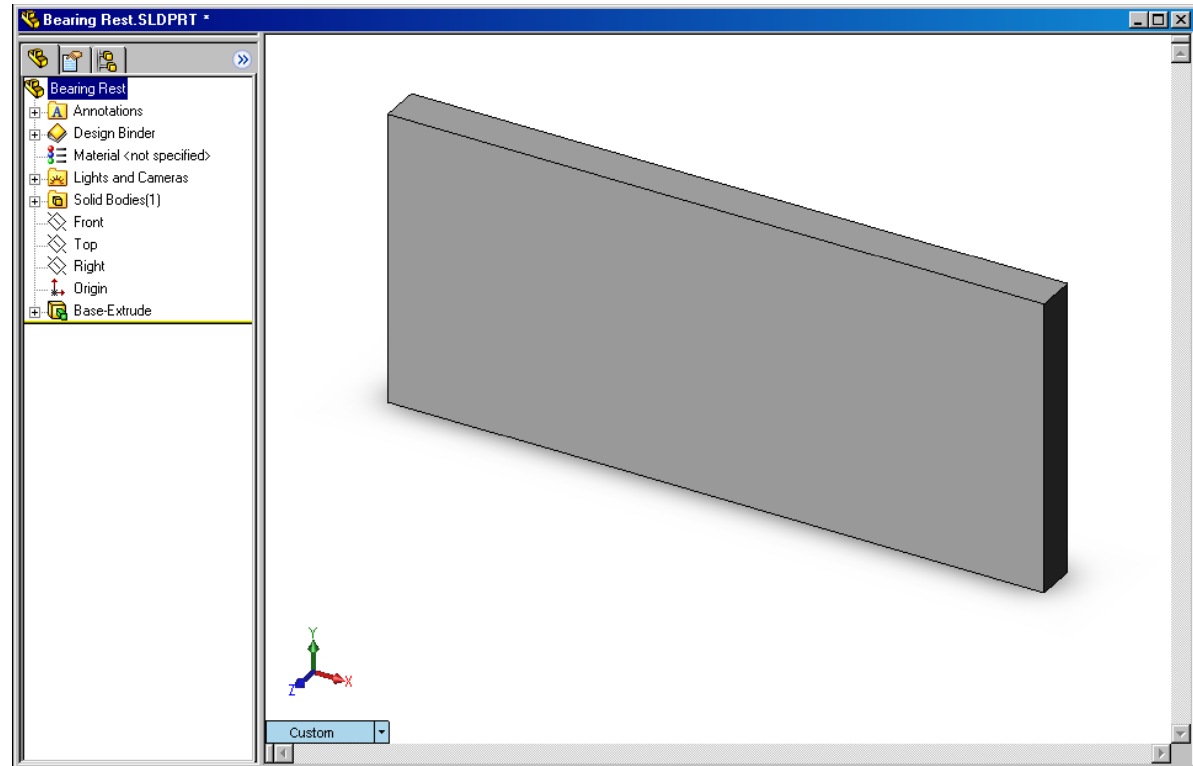
- Features are the building blocks of the part.
- Features are the *shapes* and *operations* that construct the part.



Examples of Shape Features



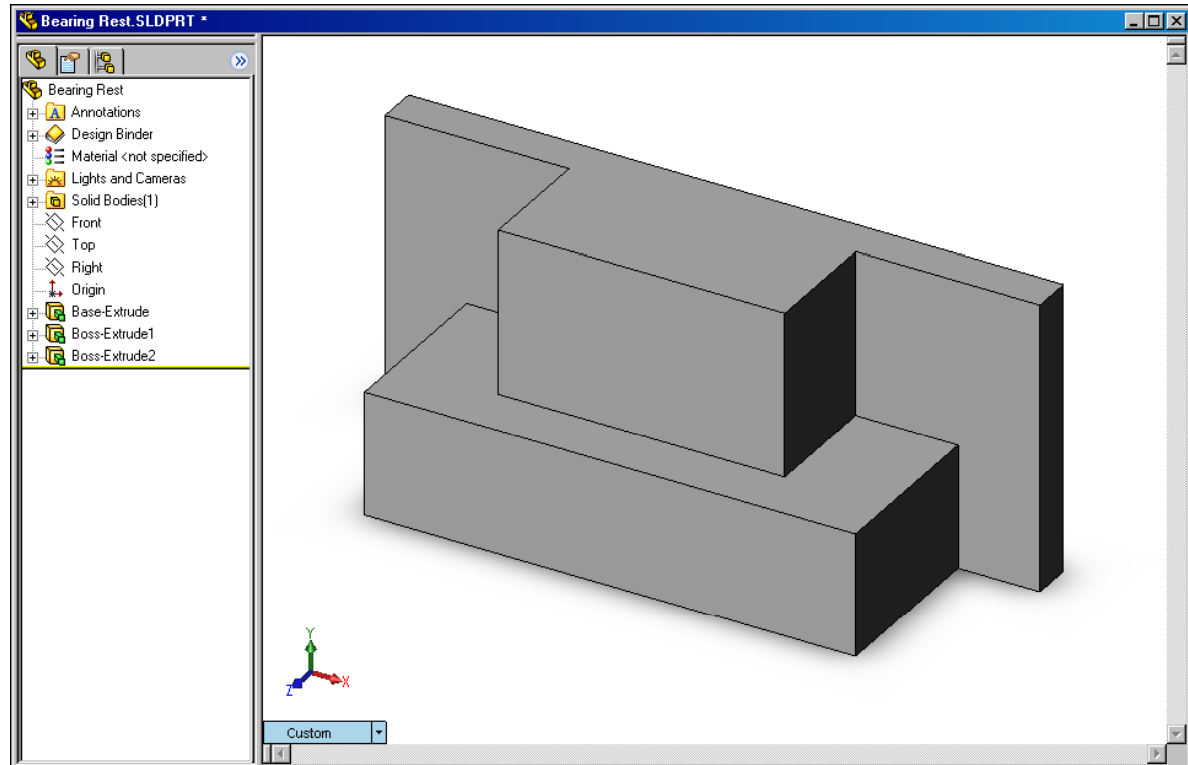
- **Base Feature**
 - First feature in part.
 - Created from a 2D sketch.
 - Forms the work piece to which other features are added.



Examples of Shape Features



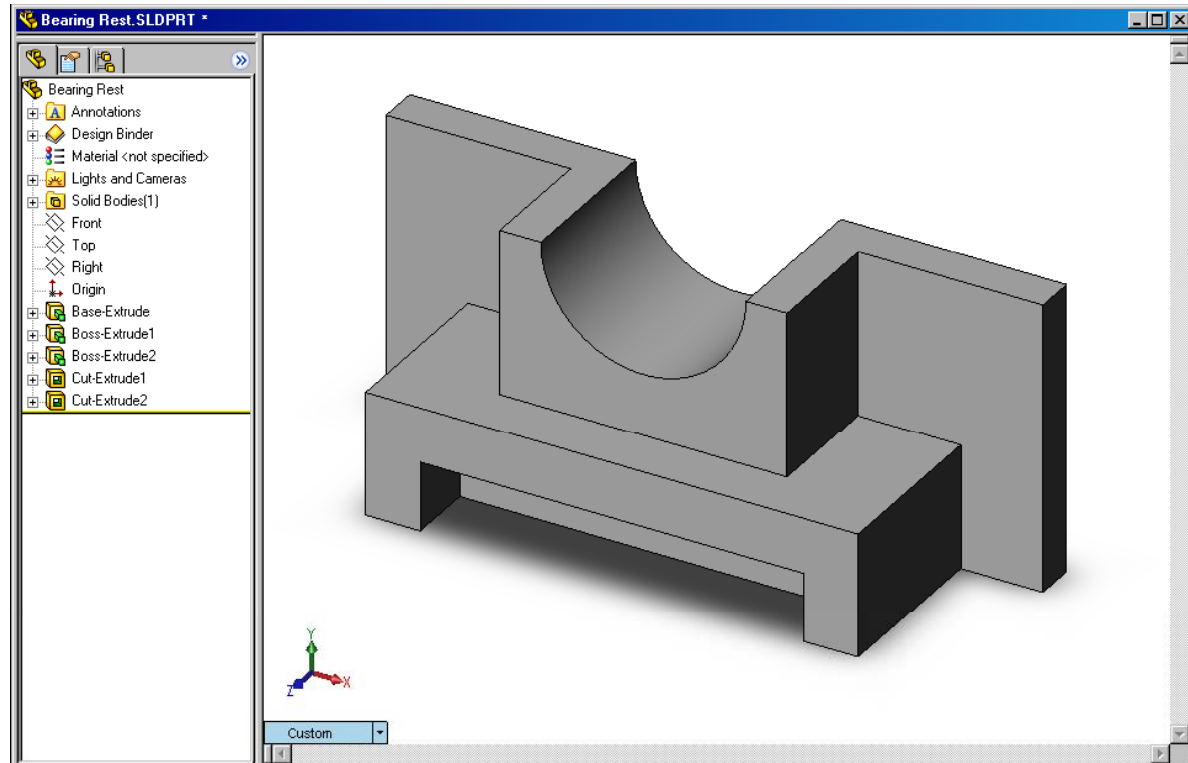
- **Boss feature**
 - Adds material to part.
 - Created from 2D sketch.



Examples of Shape Features



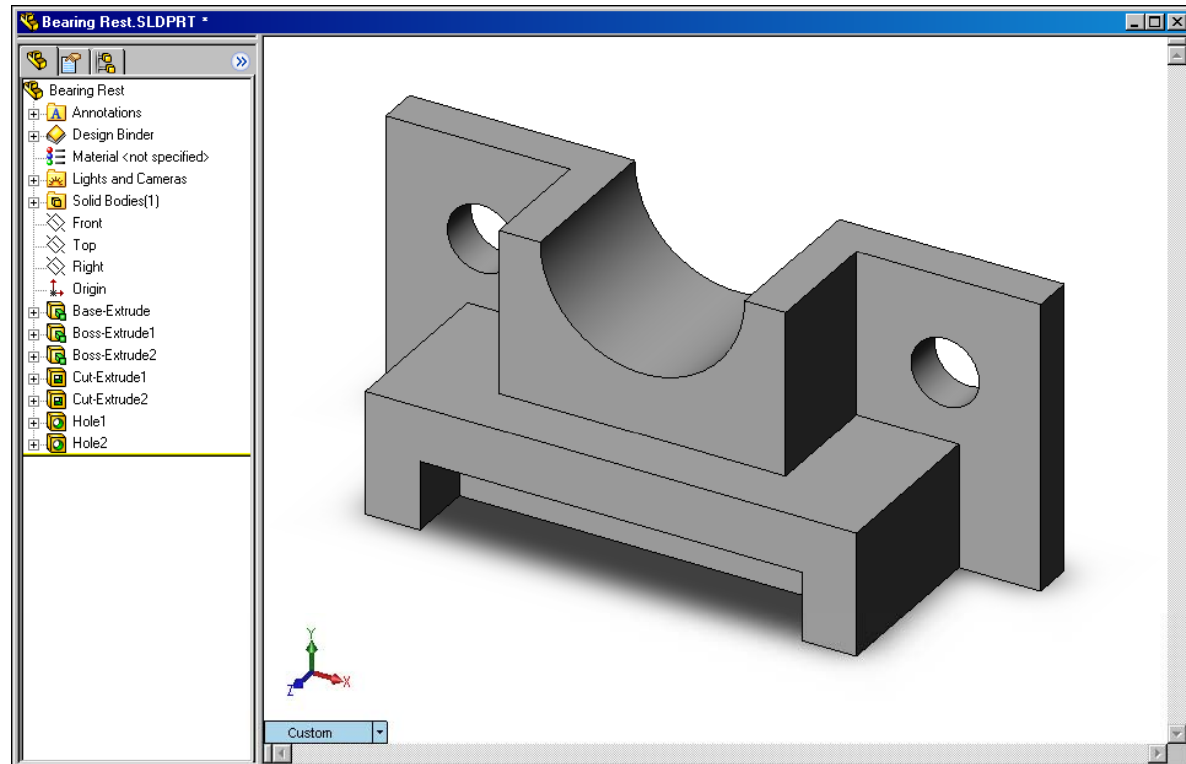
- **Cut feature**
 - Removes material from part.
 - Created from 2D sketch.



Examples of Shape Features



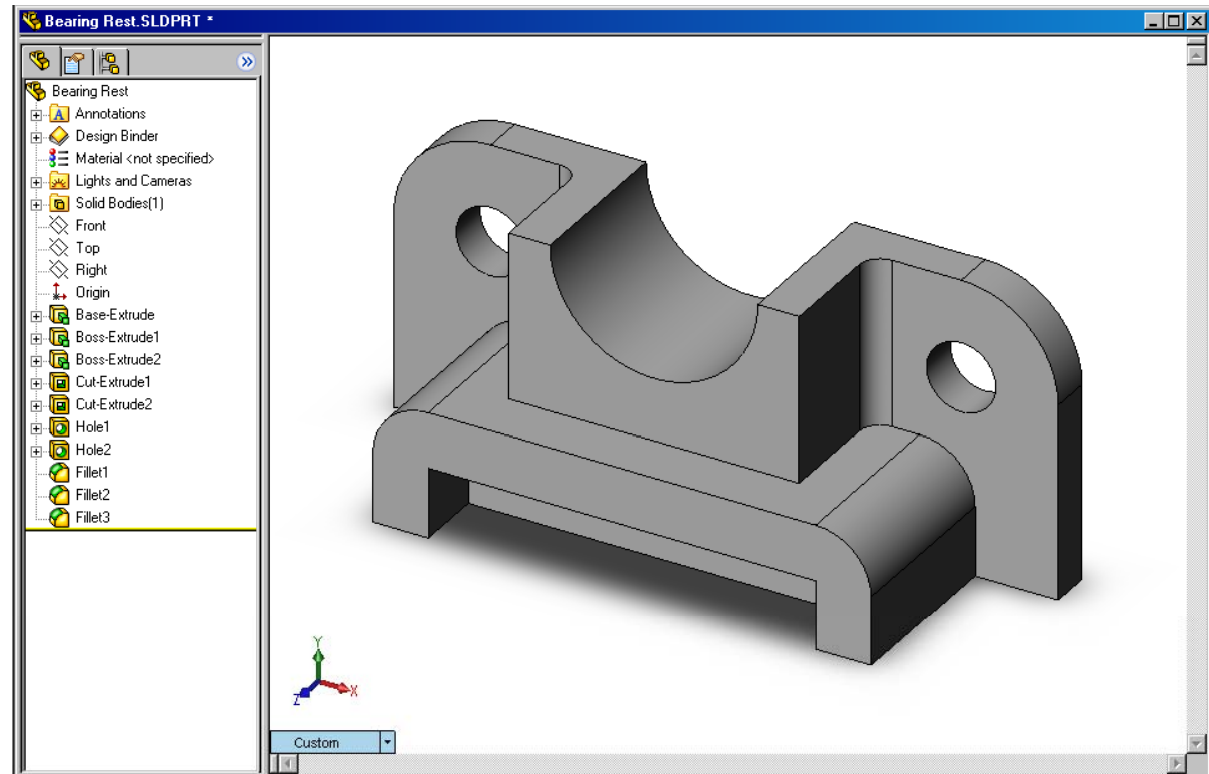
- **Hole feature**
 - Removes material.
 - Works like more intelligent cut feature.
 - Corresponds to process such as counter-sink, thread, counter-bore.



Examples of Shape Features



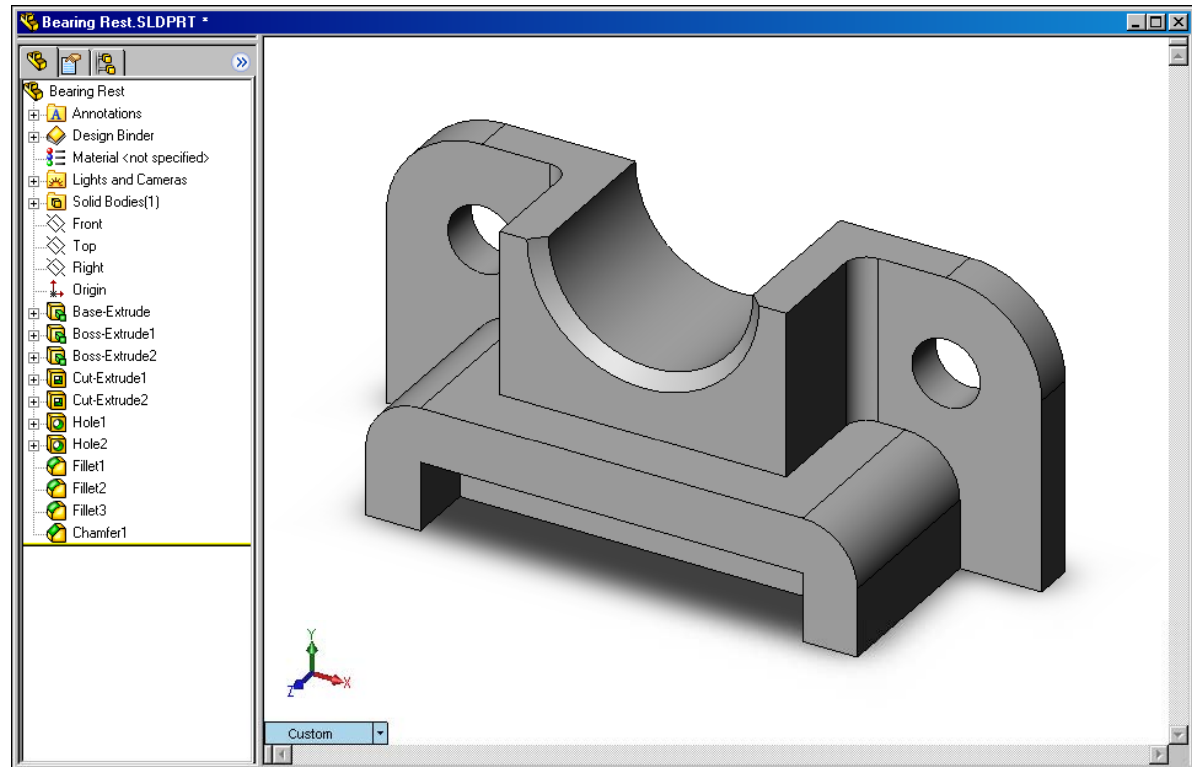
- **Fillet feature**
 - Used to round off sharp edges.
 - Can remove or add material.
 - Outside edge (convex fillet) removes material.
 - Inside edge (concave fillet) adds material.



Examples of Shape Features



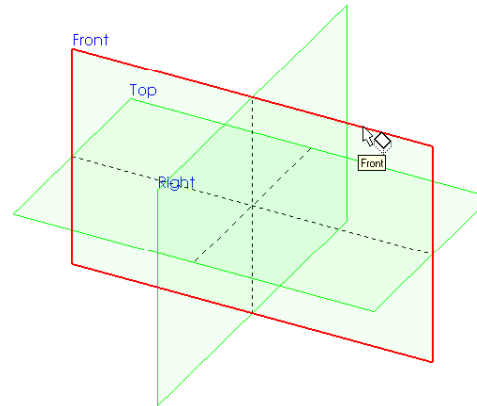
- **Chamfer feature**
 - Similar to a fillet.
 - Bevels an edge rather than rounding it.
 - Can remove or add material.



To Create an Extruded Base Feature:

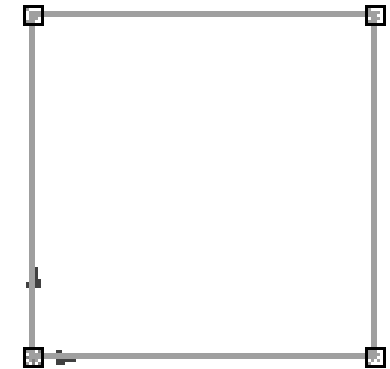


1. Select a sketch plane.



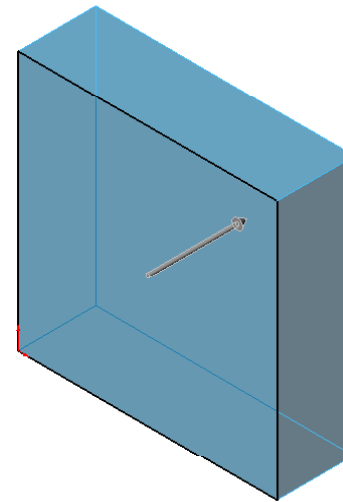
•Select the sketch plane

2. Sketch a 2D profile.

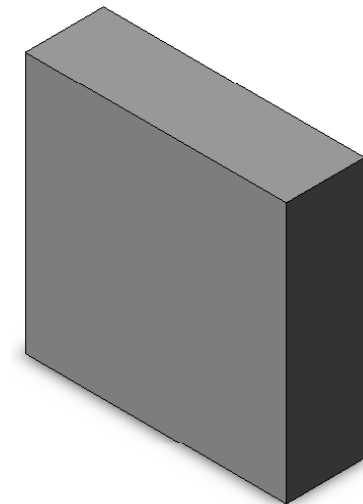


•Sketch the 2D profile

3. Extrude the sketch perpendicular to sketch plane.



•Extrude the sketch

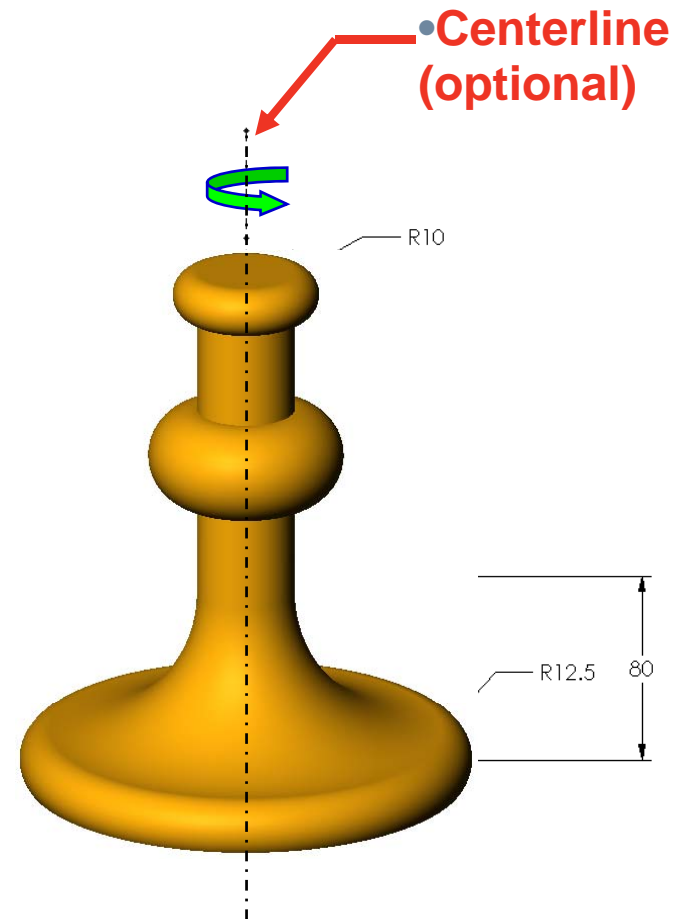


•Resulting base feature

To Create a Revolved Base Feature:



1. Select a sketch plane.
2. Sketch a 2D profile.
3. Sketch a centerline (optional).
4. Revolve the sketch around a sketch line or centerline.



Terminology: Document Window



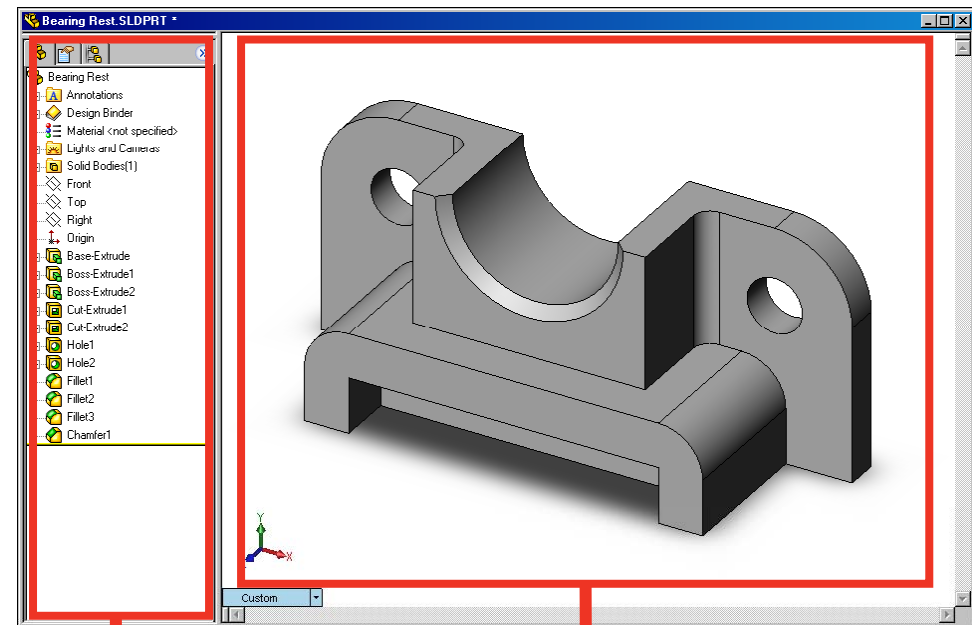
- **Divided into two panels:**

- **Left panel contains the FeatureManager® design tree.**

- Lists the structure of the part, assembly or drawing.

- **Right panel contains the Graphics Area.**

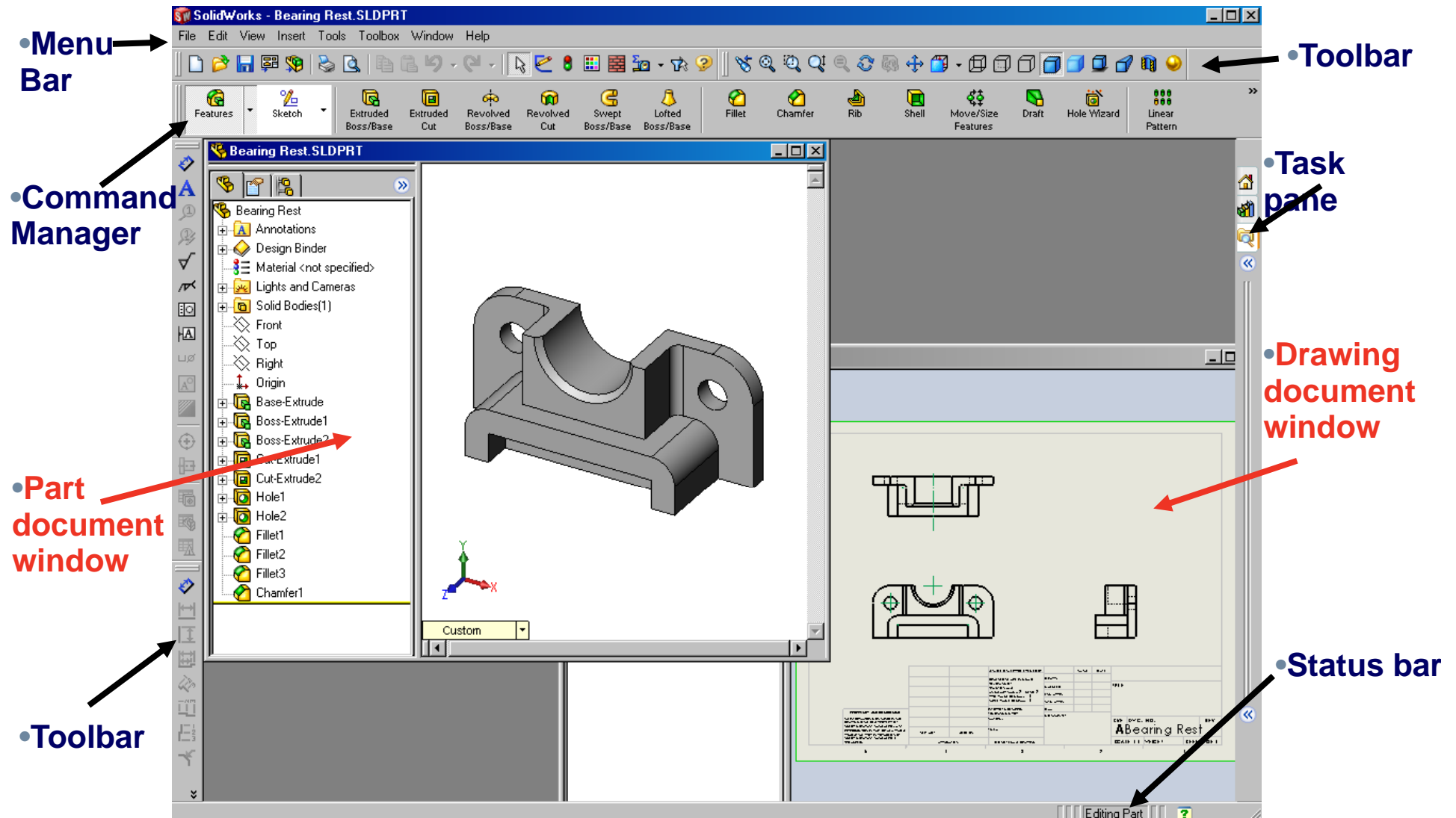
- Location to display, create, and modify a part, assembly or drawing.



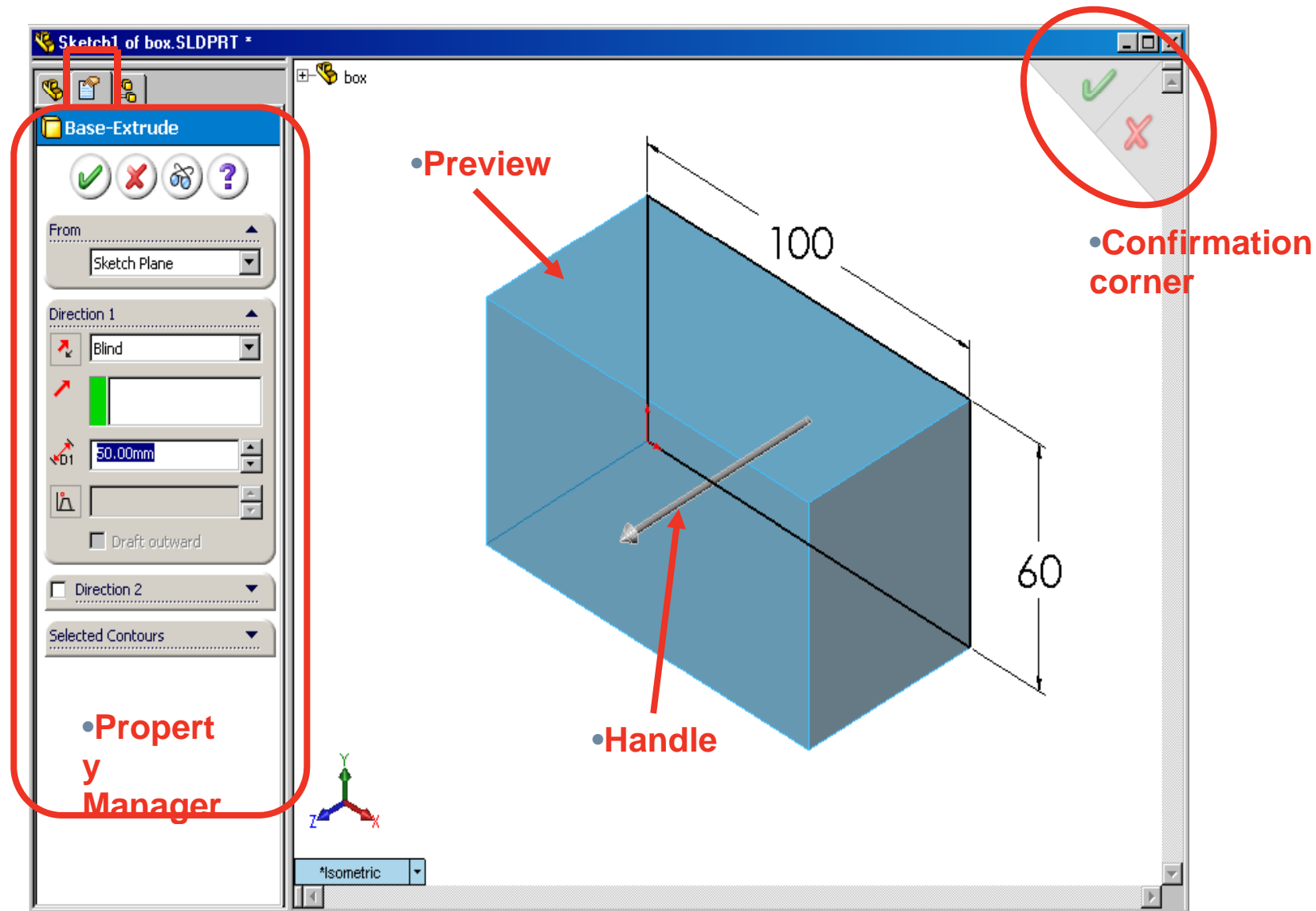
• **FeatureManager
design tree**

• **Graphics Area**

Terminology: User Interface



Terminology: PropertyManager

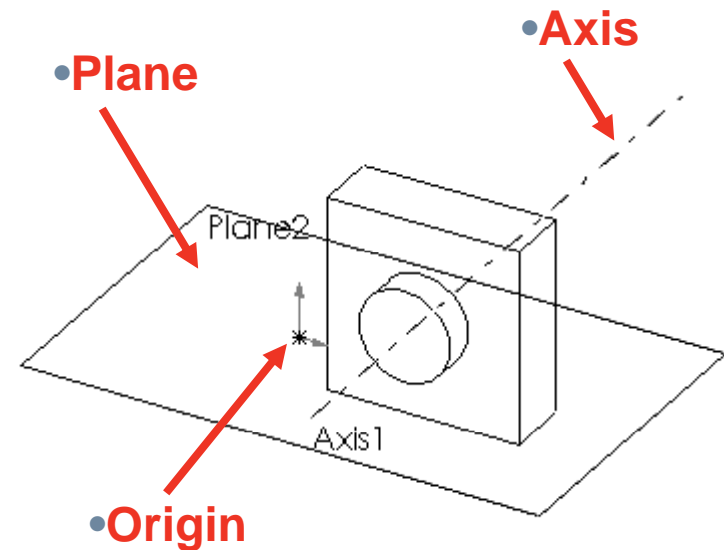


Terminology: Basic Geometry






- **Axis** - An implied centerline that runs through every cylindrical feature.
- **Plane** - A flat 2D surface.
- **Origin** - The point where the three default reference planes intersect. The coordinates of the origin are:

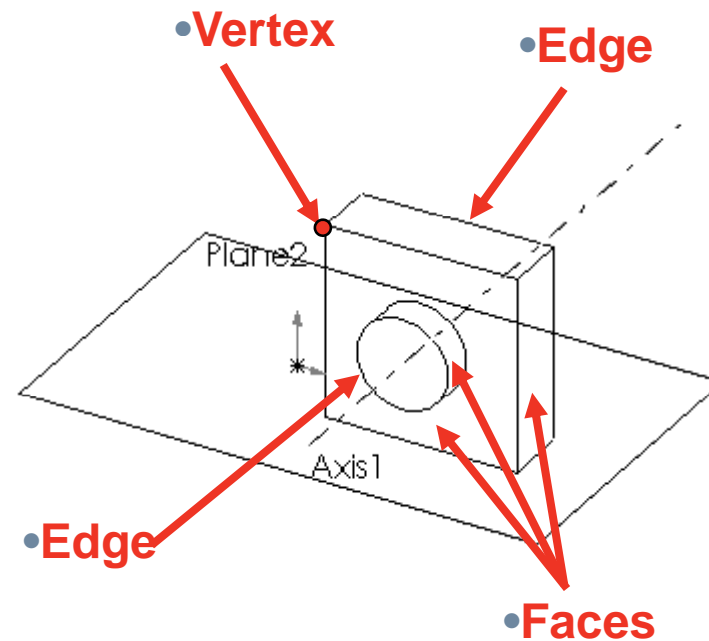
$(x = 0, y = 0, z = 0)$.



Terminology: Basic Geometry




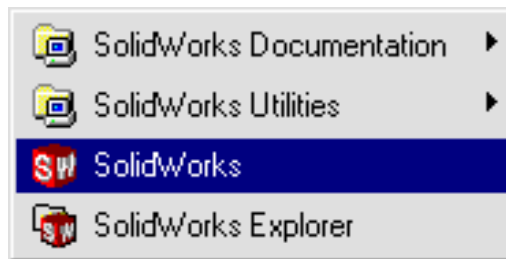
- **Face**  –
The surface or “skin” of a part. Faces can be flat or curved.
- **Edge**  –
The boundary of a face. Edges can be straight or curved.
- **Vertex**  –
The corner where edges meet.



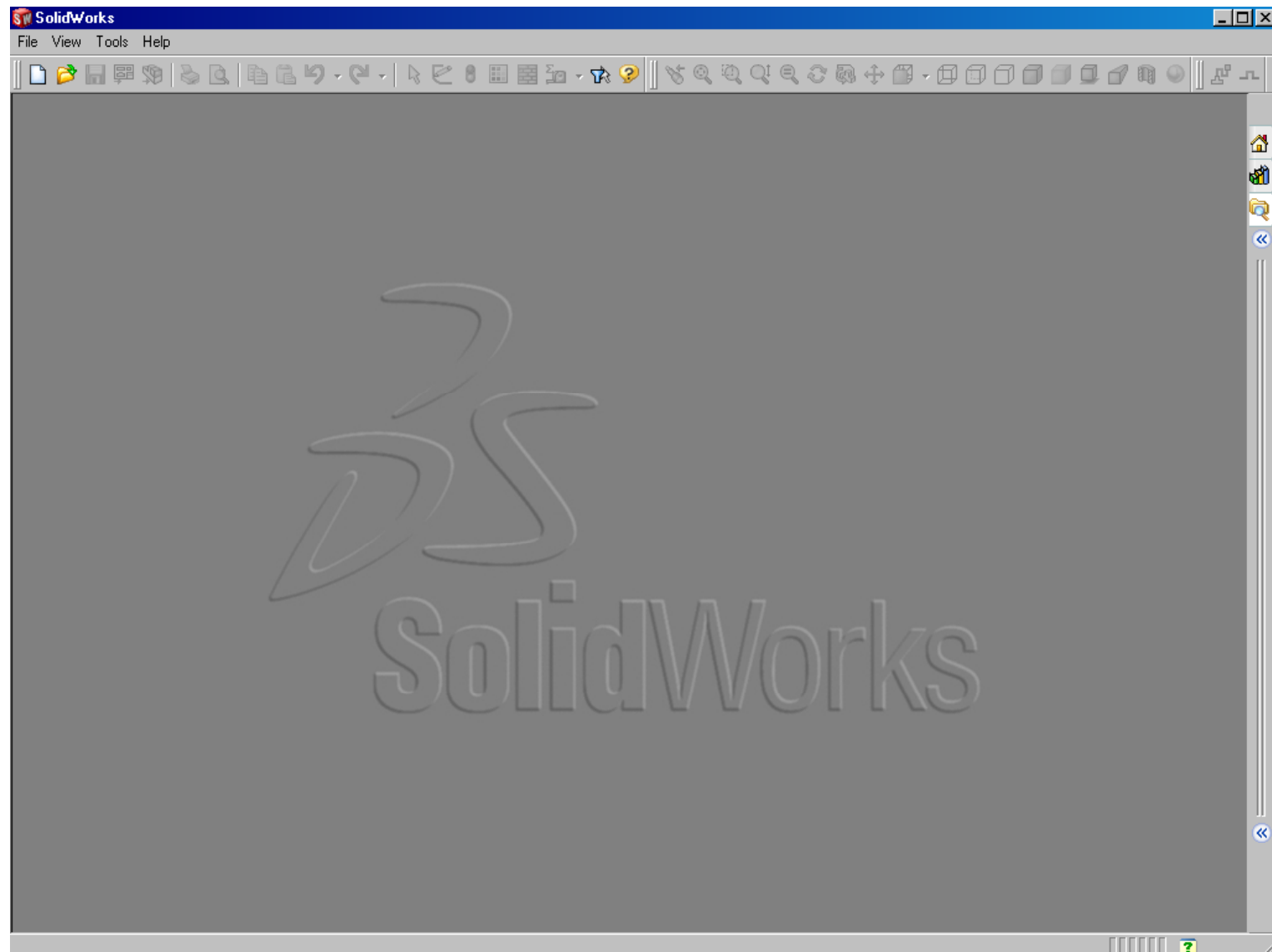
To Start SolidWorks



- Click the Start button  on Windows task bar.
- Click Programs.
- Click the SolidWorks folder.
- Click the SolidWorks application.




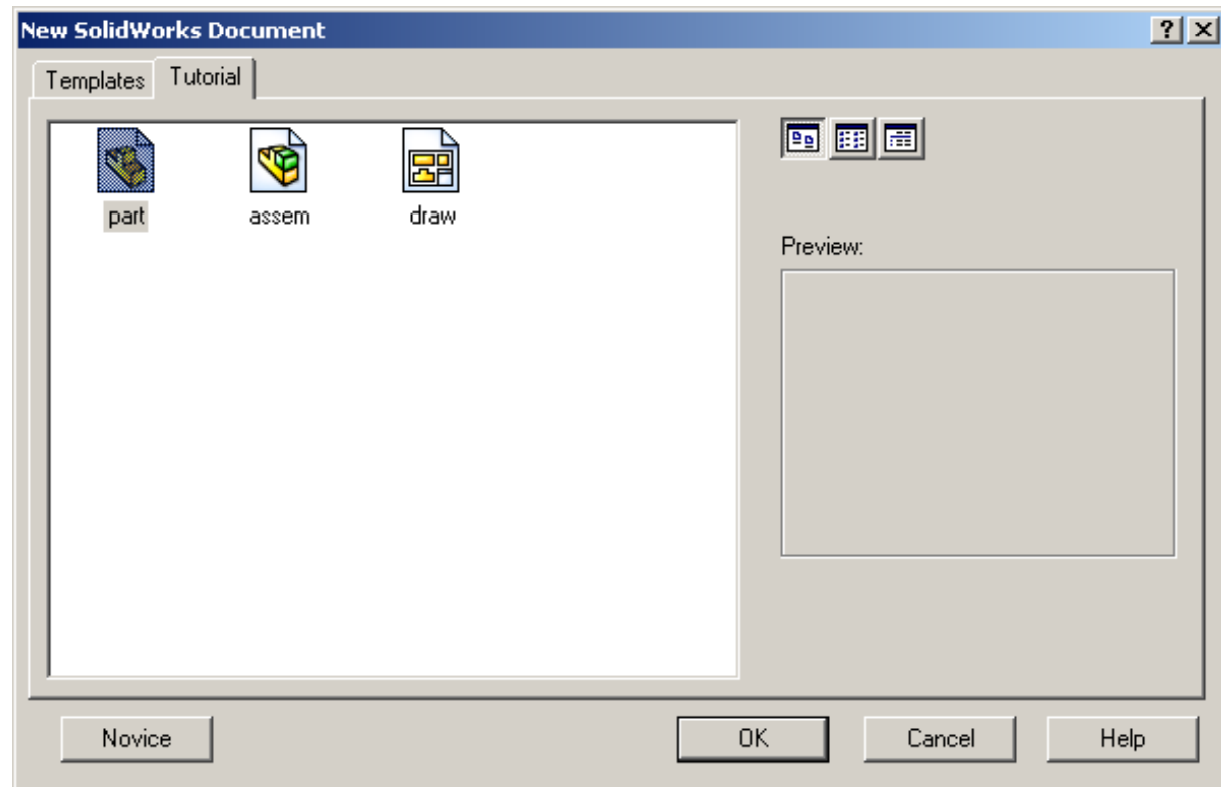
The SolidWorks Window



Creating New Files Using Templates



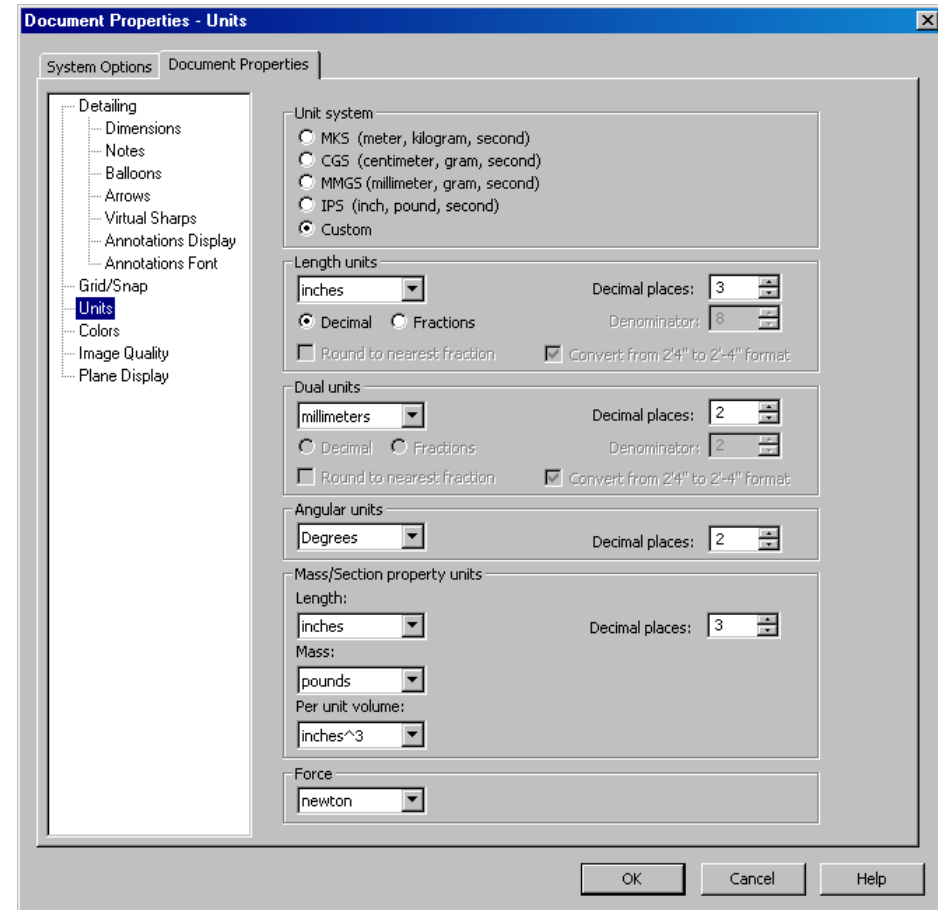
- Click **New**  on the Standard toolbar.
- Select a document template:
 - Part
 - Assembly
 - Drawing



Document Properties

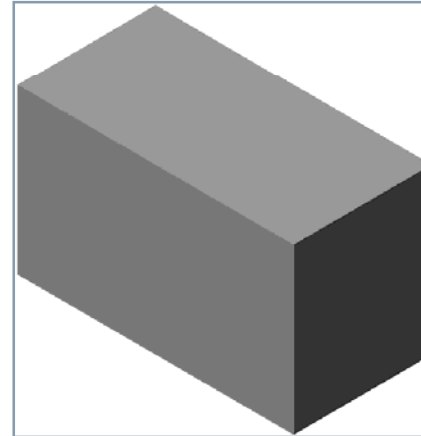


- Accessed through the Tools, Options menu.
- Control settings like:
 - Units: English (inches) or Metric (millimeters)
 - Grid/Snap Settings
 - Colors, Material Properties and Image Quality

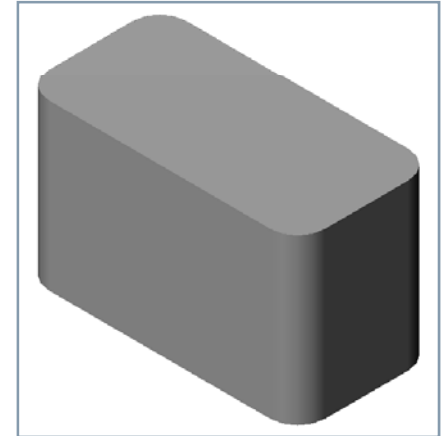


Features used to build the *box* are:

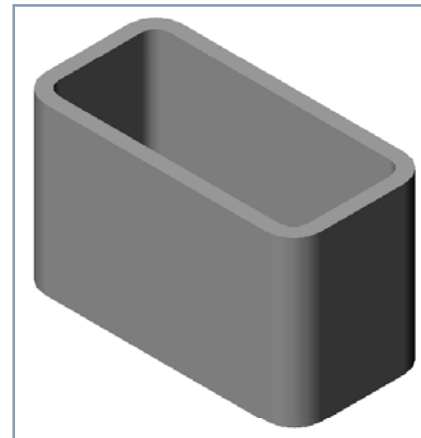
- **Extruded Base feature**
- **Fillet feature**
- **Shell feature**
- **Extruded Cut feature**



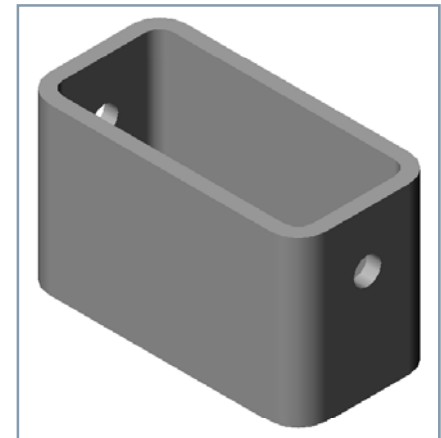
•1.Base Feature



•2.Fillet Feature





•3.Shell Feature

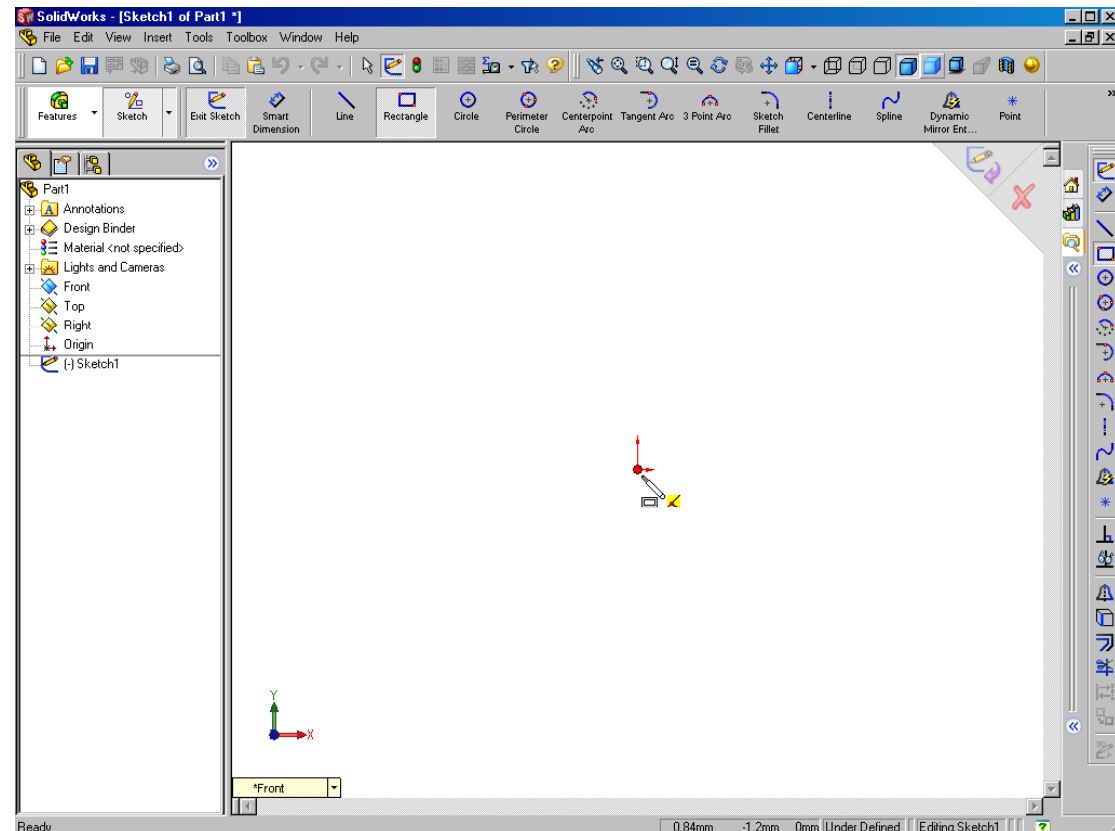


•4.Cut Feature

Creating a 2D Sketch



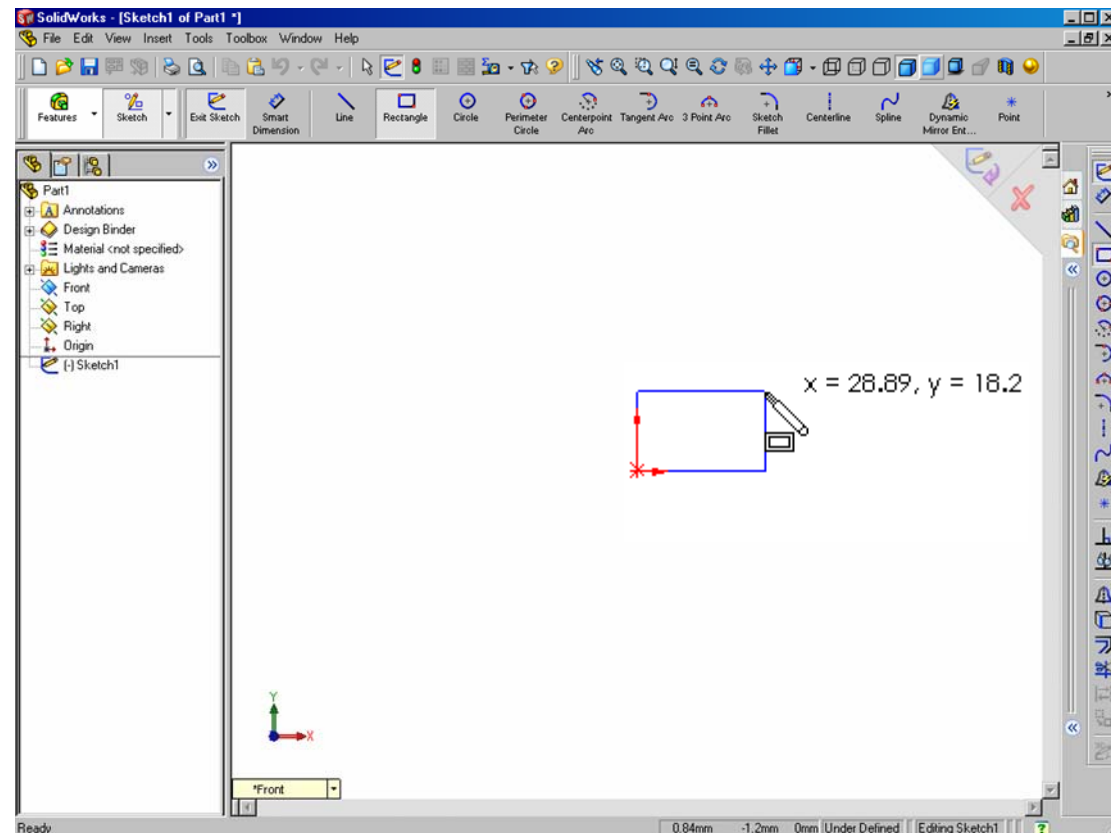
1. Click Sketch  on the Sketch toolbar.
2. Select the Front plane as a sketch plane.
3. Click Rectangle  on the Sketch Tools toolbar.
4. Move the pointer to the Sketch Origin.



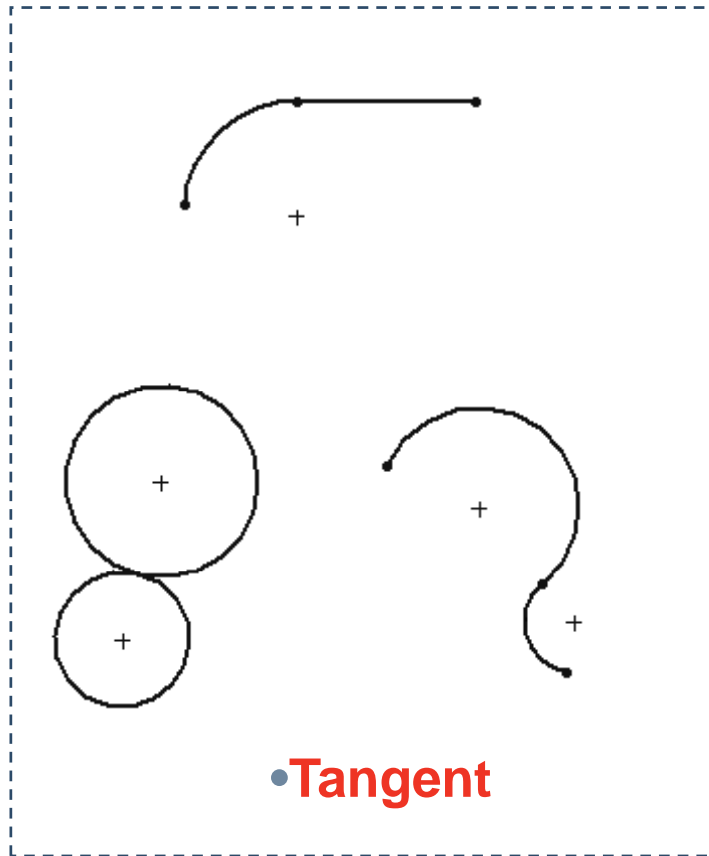
Creating a 2D Sketch



5. Click the left mouse button.
6. Drag the pointer up and to the right.
7. Click the left mouse button again.



Geometric Relationships



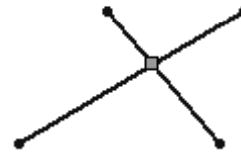
•Tangent



•Vertical



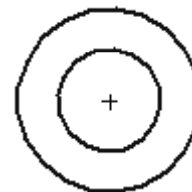
•Horizontal



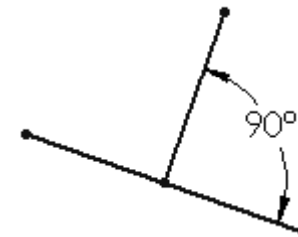
•Intersection



•Parallel



•Concentric




•Perpendicular

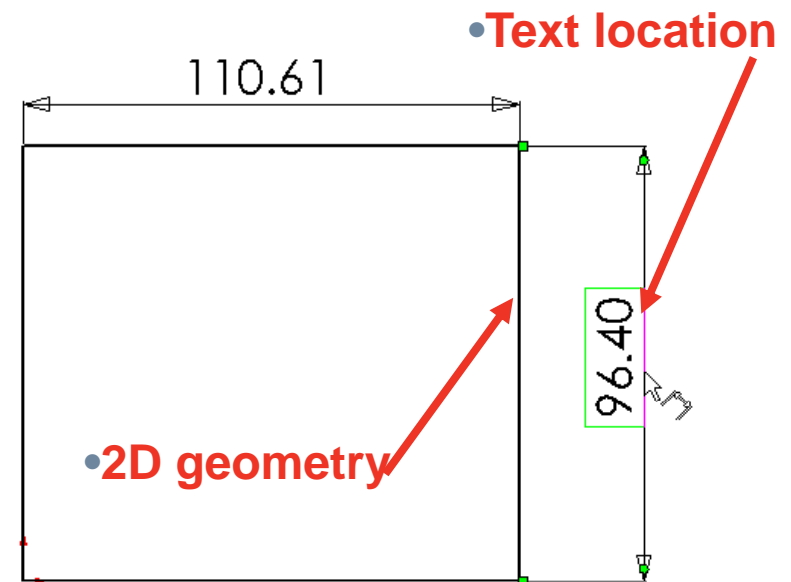
Adding Dimensions



- **Dimensions specify the size of the model.**

To create a dimension:

1. Click Dimension  on the Sketch Relations toolbar.
2. Click the 2D geometry.
3. Click the text location.
4. Enter the dimension value.

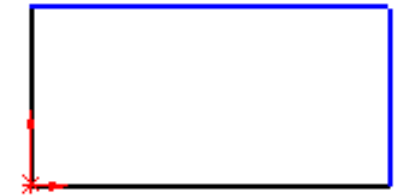


The Status of a Sketch



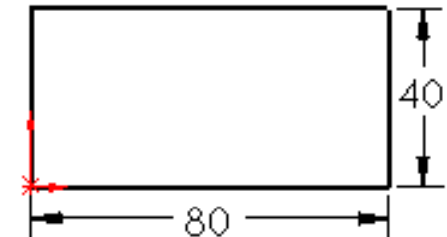
- **Under defined**

- Additional dimensions or relations are required.
- Under defined sketch entities are *blue* (by default).



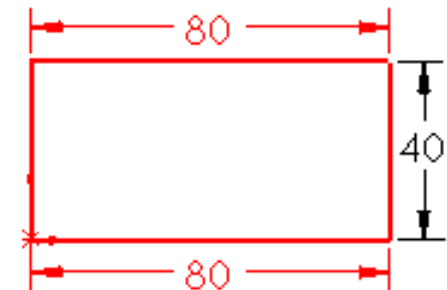
- **Fully defined**

- No additional dimensions or relationships are required.
- Fully defined sketch entities are *black* (by default).



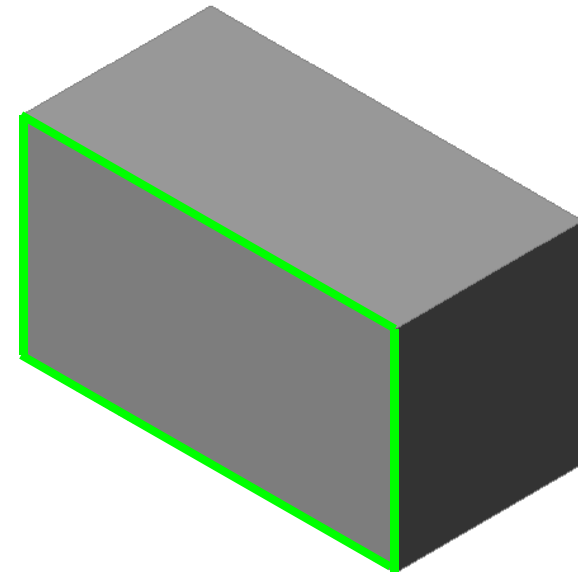
- **Over defined**

- Contains conflicting dimensions or relations, or both.
- Over defined sketch entities are *red* (by default).



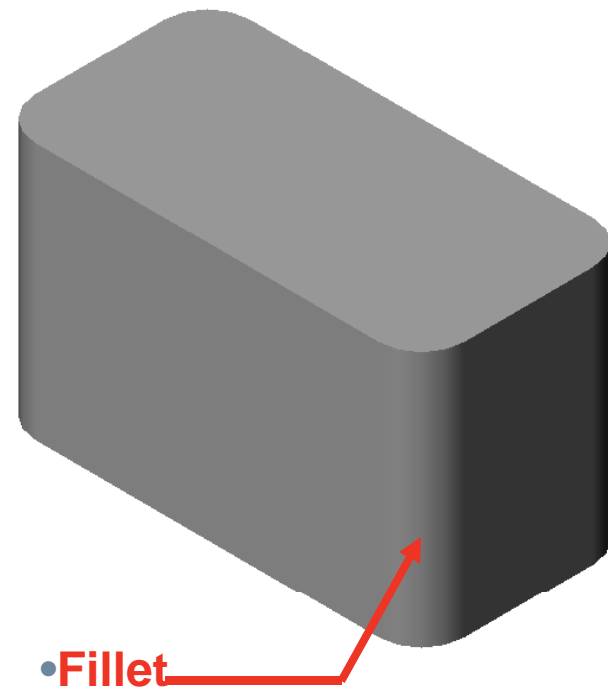
To create the extruded base feature for the *box*:

- **Sketch a rectangular profile on a 2D plane.**
- **Extrude the sketch.**
- **By default extrusions are perpendicular to the sketch plane.**



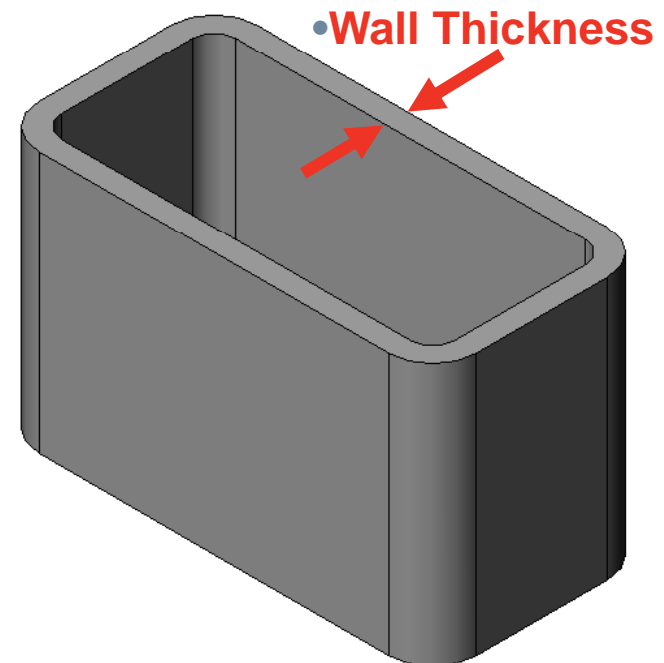
Fillet feature

- The fillet feature rounds the edges or faces of a part.
- Select the edges to be rounded. Selecting a face rounds all the edges of that face.
- Specify the fillet radius.



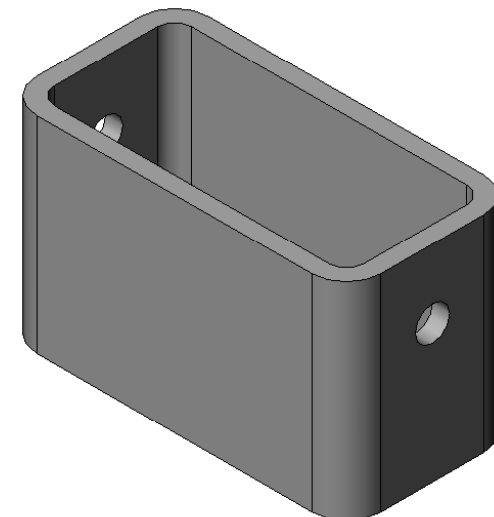
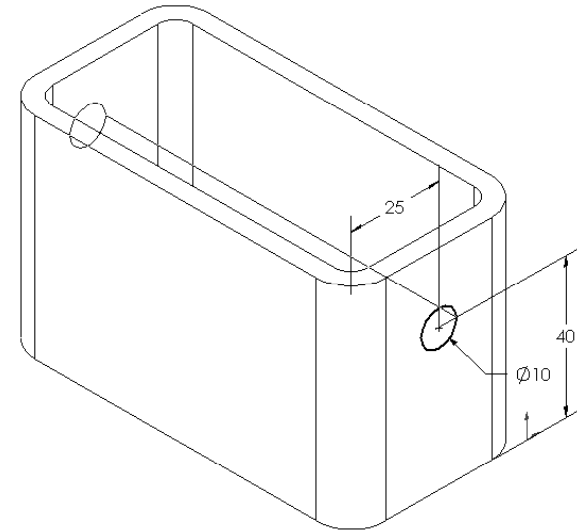
Shell feature

- The shell feature removes material from the selected face.
- Using the shell feature creates a hollow box from a solid box.
- Specify the wall thickness for the shell feature.



To create the extruded cut feature for the *box*:

- **Sketch the 2D circular profile.**
- **Extrude the 2D Sketch profile perpendicular to the sketch plane.**
- **Enter Through All for the end condition.**
- **The cut penetrates through the entire part.**



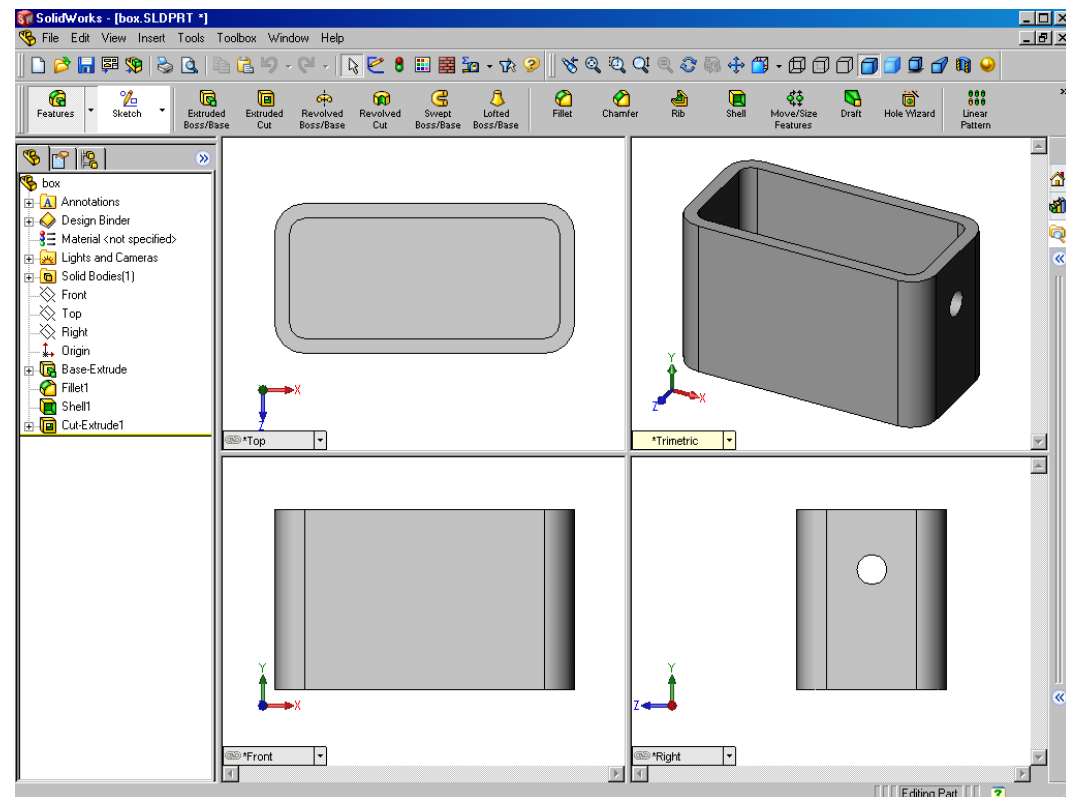
Dimensions and Geometric Relationships

- **Specify dimensions and geometric relationships between features and sketches.**
- **Dimensions change the size and shape of the part.**
- **Mathematical relationships between dimensions can be controlled by equations.**
- **Geometric relationships are the rules that control the behavior of sketch geometry.**
- **Geometric relationships help capture design intent.**

Multiple Views of a Document



- Click the view pop-up menu.
- Select an icon.
The viewport icons include:
 - Single View
 - Two View (horizontal and vertical)
 - Four View



Base Feature

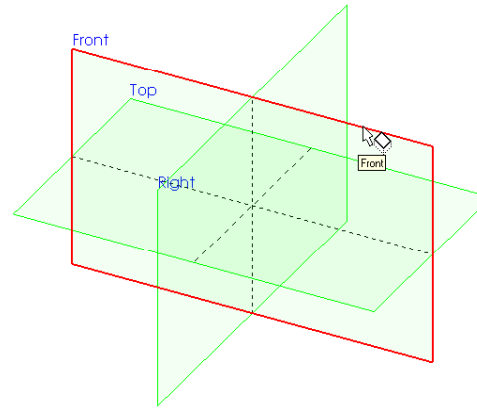
- The first feature that is created.
- The foundation of the part.
- The base feature geometry for the box is an extrusion.
- The extrusion is named *Extrude1*.

Tip: Keep the base feature simple.

To Create an Extruded Base Feature:

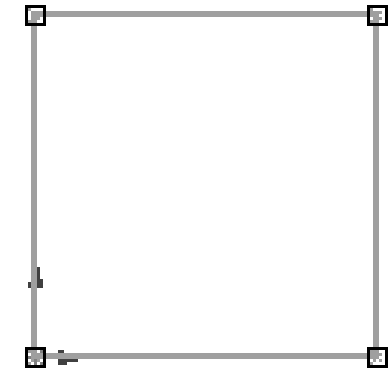


1. Select a sketch plane.



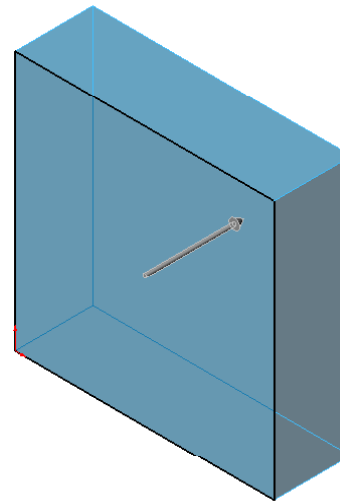
•Select the sketch plane

2. Sketch a 2D profile.

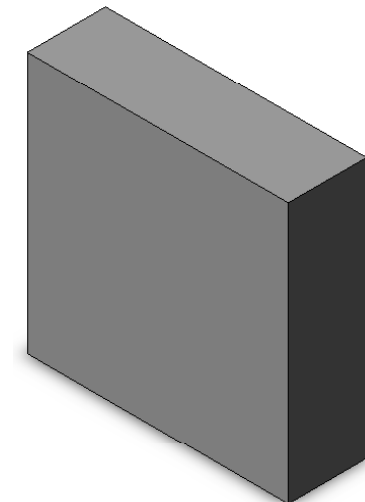


•Sketch the 2D profile

3. Extrude the sketch perpendicular to sketch plane.



•Extrude the sketch

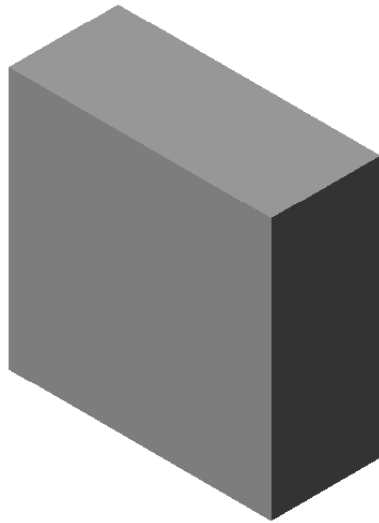


•Resulting base feature

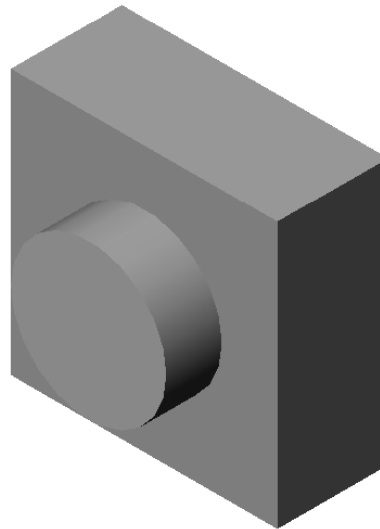
Features Used to Build Tutor1



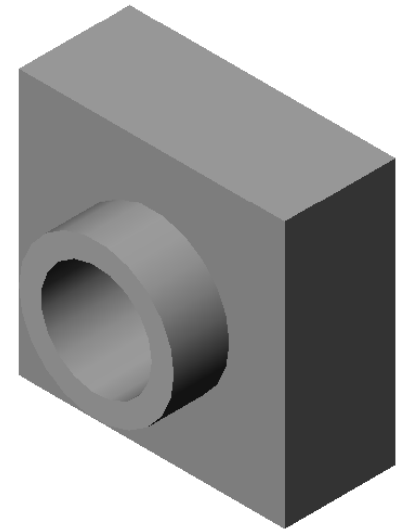
•1.Base Extrude



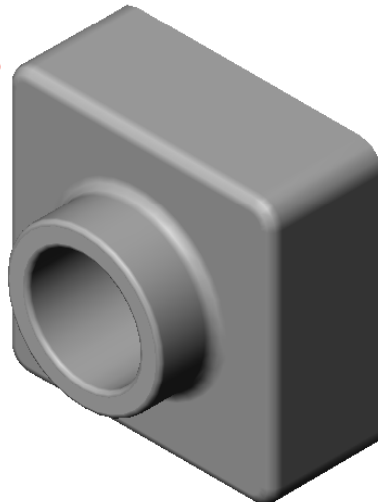
•2.Boss Extrude



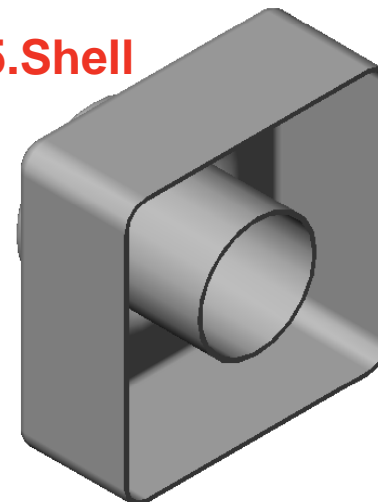
•3.Cut Extrude



•4.Fillets

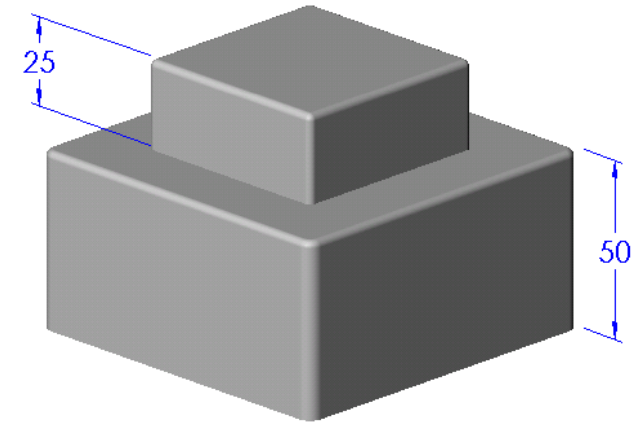


•5.Shell



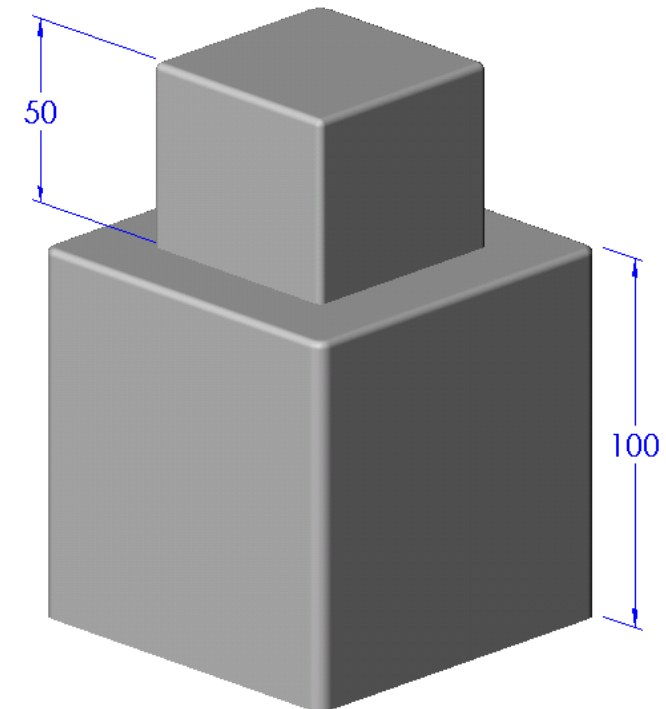
- **Dimensions**

- **Base depth = 50 mm**
- **Boss depth = 25 mm**



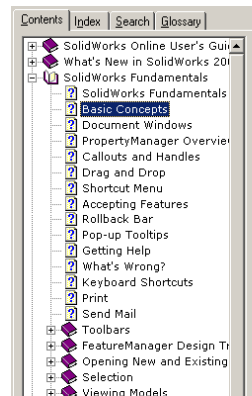
- **Mathematical relationship**

- **Boss depth = Base depth \div 2**



To view comprehensive online help:

- Click .
- Select Help, SolidWorks Help Topics.
- Help displays in a separate window.



Basic Concepts

- A SolidWorks model consists of parts, assemblies, and drawings.
- Typically, you begin with a sketch, create a base feature, and then add more features to your model. (You can also begin with an imported surface or solid geometry.)
- You are free to refine your design by adding, changing, or reordering features.
- Associativity between parts, assemblies, and drawings assures that changes made to one view are automatically made to all other views.
- You can generate drawings or assemblies at any time in the design process.
- The SolidWorks software lets you customize functionality to suit your needs.
- Click **Tools, Options** on the main menu to display the available **System Options** and **Document Properties** tabs.
- The SolidWorks software saves your work for you.